

Double-Down Boner Boost

Because two are
better than one



Important notes

If you're using any of our footswitch daughterboards, DOWNLOAD THE DAUGHTERBOARD DOCUMENT

- Download and read the appropriate build document for the daughterboard as well as this one BEFORE you start.
- DO NOT solder the supplied Current Limiting Resistor (CLR) to the main circuit board even if there is a place for it. This should be soldered to the footswitch daughterboard.

POWER SUPPLY

Unless otherwise stated in this document this circuit is designed to be powered with 9V DC.

COMPONENT SPECS

Unless otherwise stated in this document:

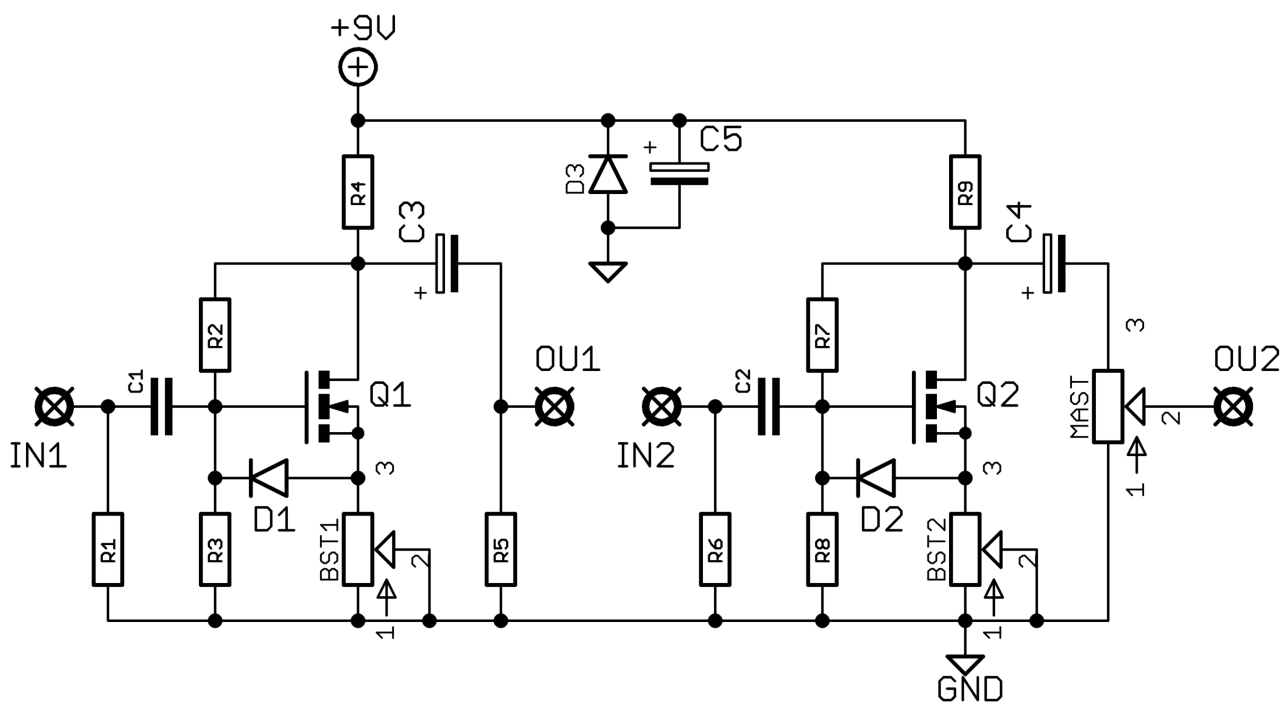
- Resistors should be 0.25W. You can use those with higher ratings but check the physical size of them.
- Electrolytics caps should be at least 25V for 9V circuits, 35V for 18V circuits. Again, check physical size if using higher ratings.

LAYOUT CONVENTIONS

Unless otherwise stated in this document, the following are used:

- **Electrolytic capacitors:**
Long leg (anode) to square pad.
- **Diodes/LEDs:**
Striped leg (cathode) to square pad. Short leg to square pad for LEDs.
- **ICs:**
Square pad indicates pin 1.

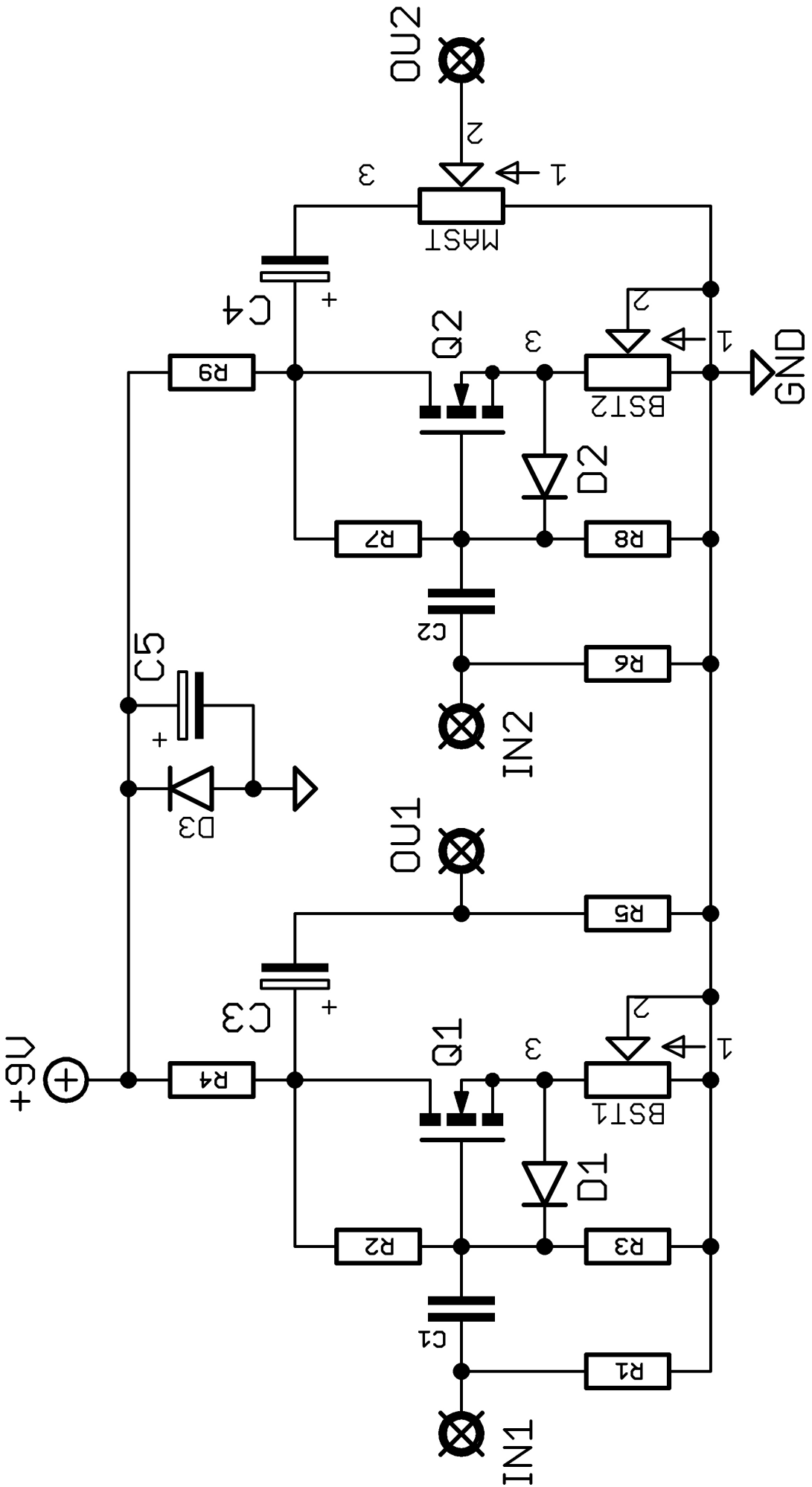
Schematic + BOM

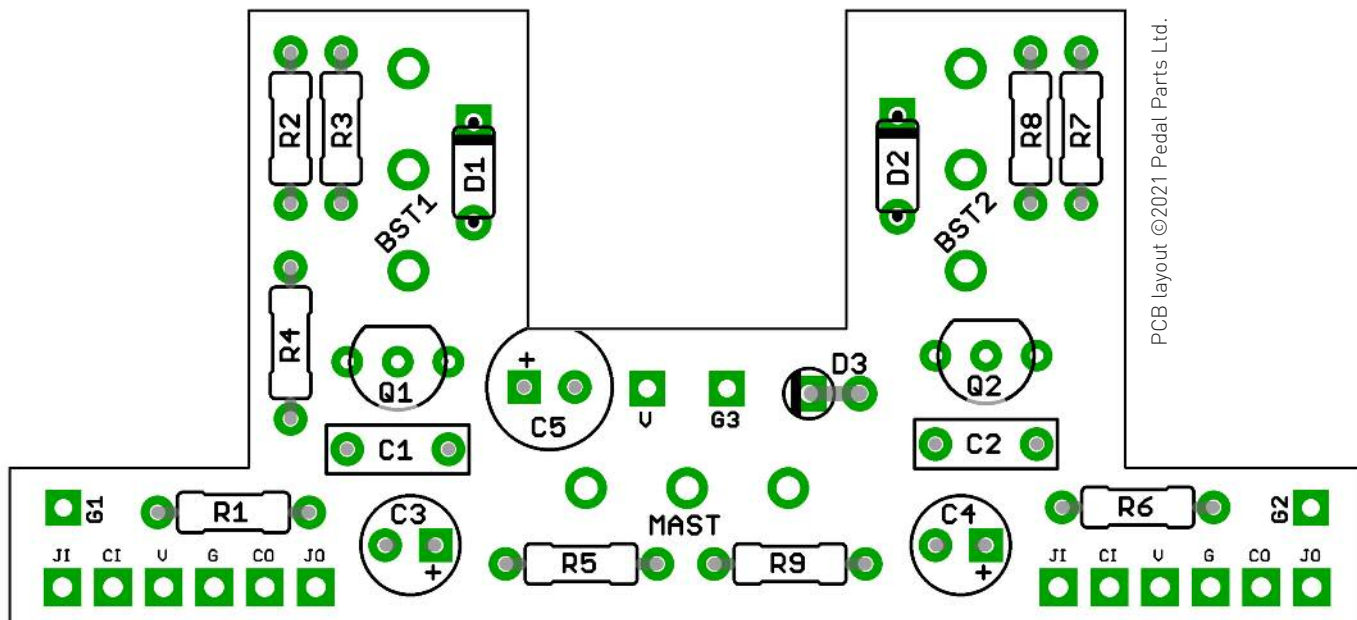


R1	1M*	C1	100n	D1-2	9V1 zener
R2	10M	C2	100n	D3	1N4001
R3	10M	C3	10u elec	BST1	5KC
R4	5K1	C4	10u elec	BST2	5KC
R5	100K	C5	100u elec	MAST	50KB
R6	1M*	Q1-2	BS170		
R7	10M				
R8	10M				
R9	5K1				

This is essentially two Boner Boosts in series with a master volume on the second. They are independent of each other, so you can use the first on it's own, the second on it's own with the master volume, or both in series, boost 1 pushing boost 2 with the resulting oomph kept in check with the master.

*Optional anti-pop pulldown resistors.





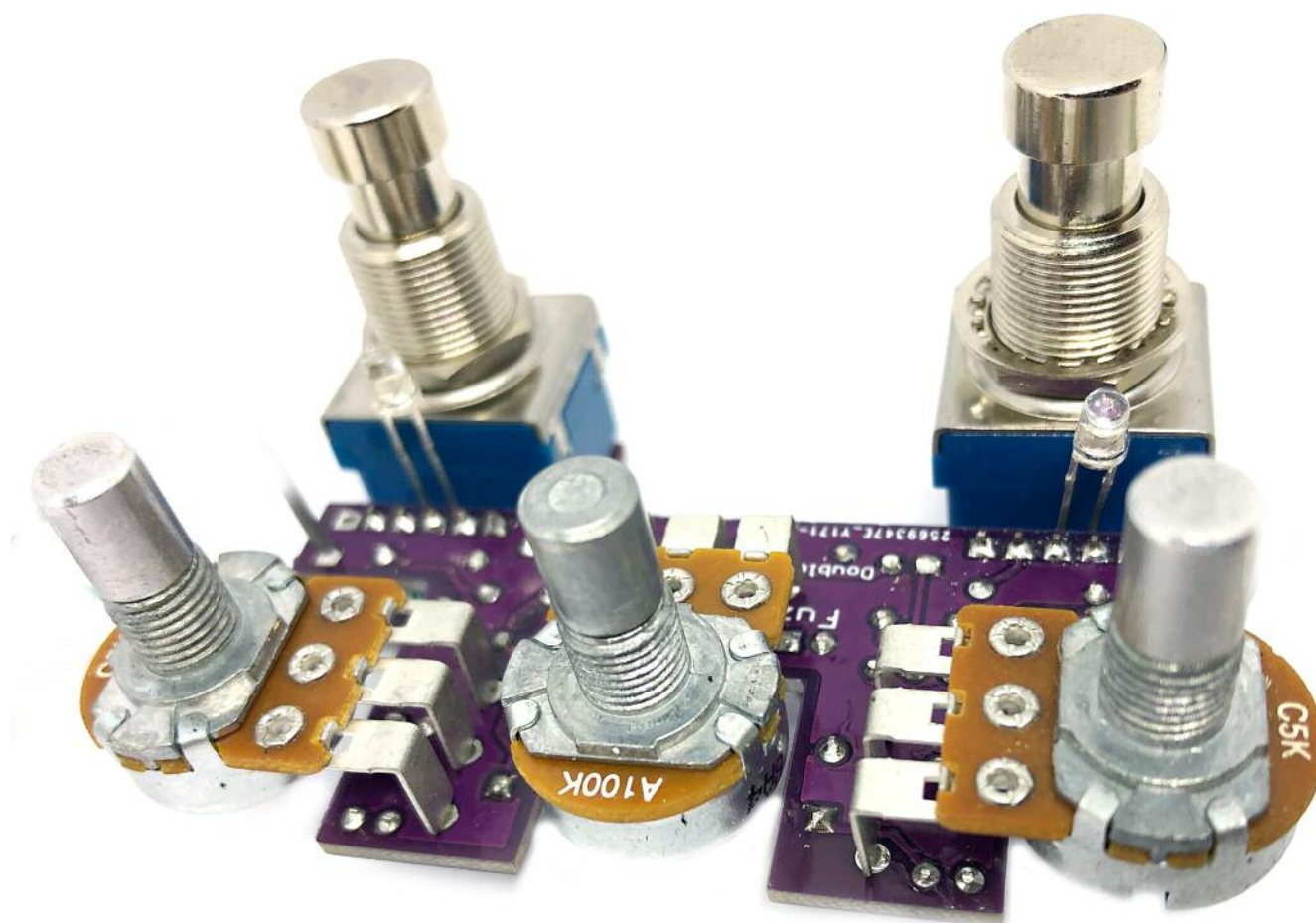
Though the power and signal pads on the PCB conform to the FuzzDog Direct Connection format, you should only connect five of the pads on each side. See next page for details.

Be very careful when soldering the diodes, transistors and LEDs. They're very sensitive to heat. You should use some kind of heat sink (crocodile clip or reverse action tweezers) on each leg as you solder them. Keep exposure to heat to a minimum (under 2 seconds).

Extra care should be taken with the BS170s. They are incredibly sensitive to static. Ensure you're not buzzing with charge before handing them or they'll simply fry.

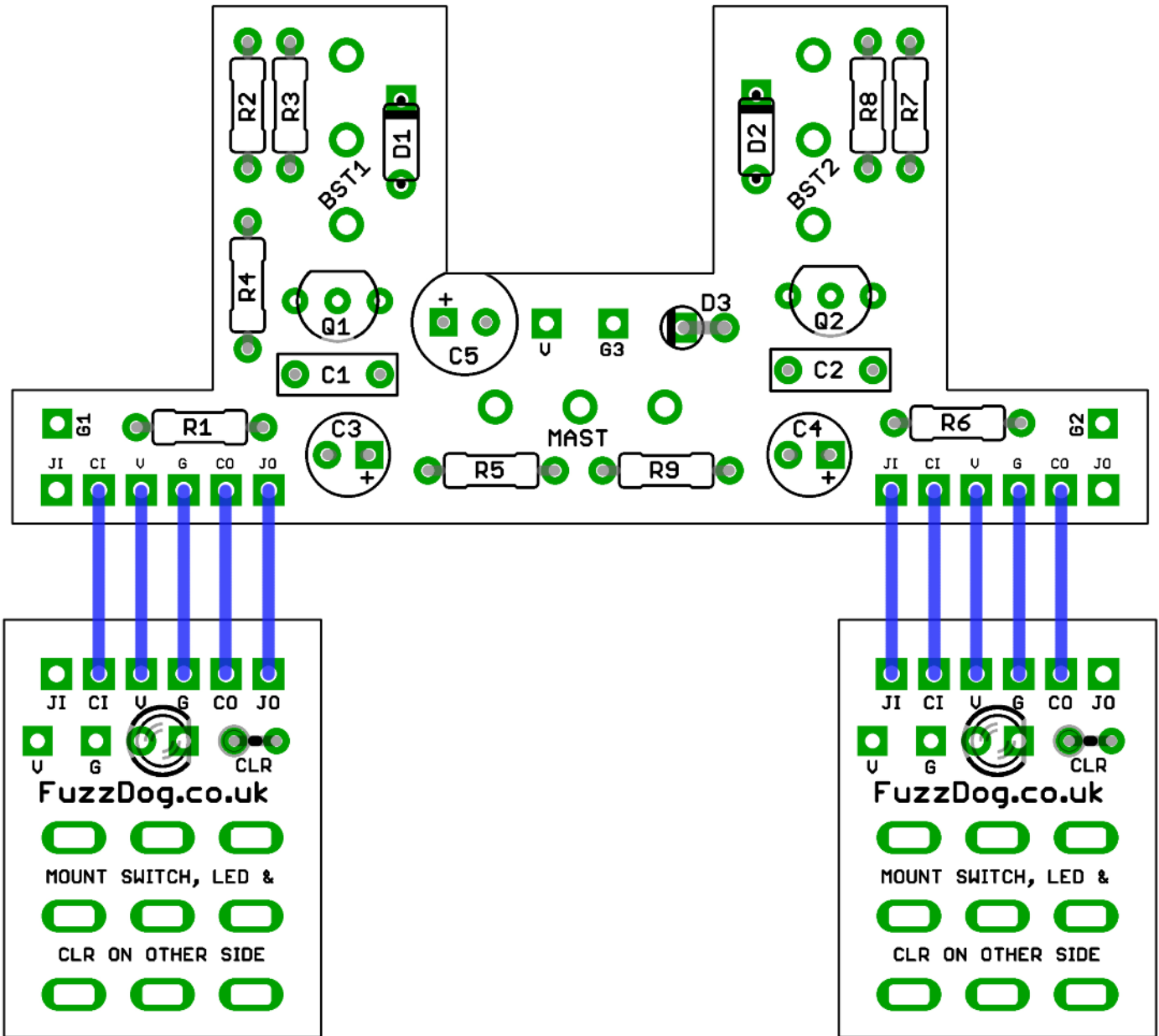
Snap the small metal tag off the pots so they can be mounted flush in the box.

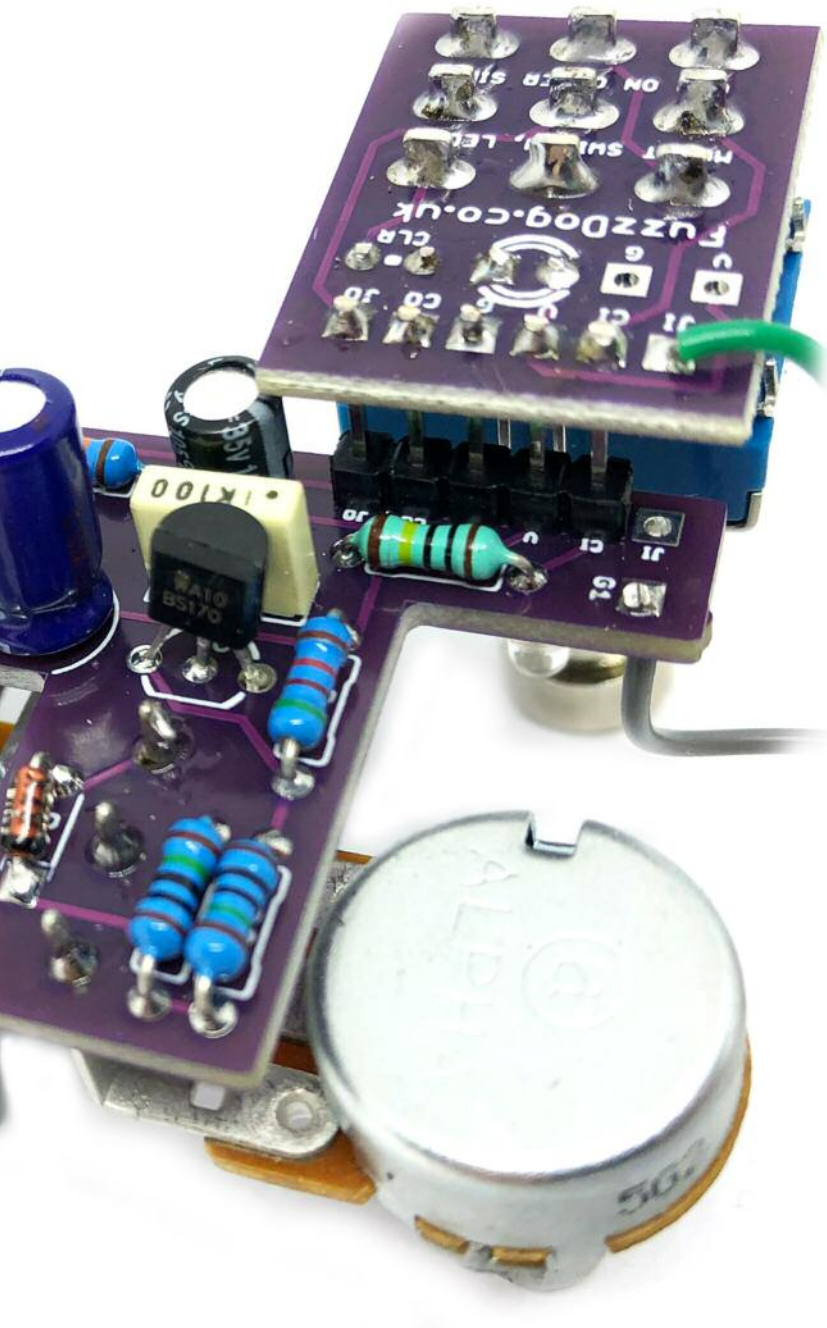
You should solder all other board-mounted components before you solder the pots. Once they're in place you'll have no access to much of the board. Make sure your pots all line up nicely.



Connecting the boards

Whether you're using header pins in a 1590B build, or wires/ribbons in a bigger box, you should only connect the ten pads shown below. The free JI and JO pads on the daughterboards will be used to connect the IN and OUT jacks.





Joining with headers

This method requires precise enclosure drilling. To assemble like this, solder your 5-pin headers into the main PCB. Ensure they are mounted absolutely vertical.

Mount the pots into their holes in the enclosure, but don't fully tighten.

Mount your footswitches loosely into the enclosure. Ensure you have the inner nut a few turns down the thread away from the switch body. This is to ensure there's enough clearance between the two PCBs for the 10u caps - we'll move those across a bit on the next board revision.

Pull your LEDs through the daughterboard before you drop it onto the headers and switch, otherwise you'll never get them in.

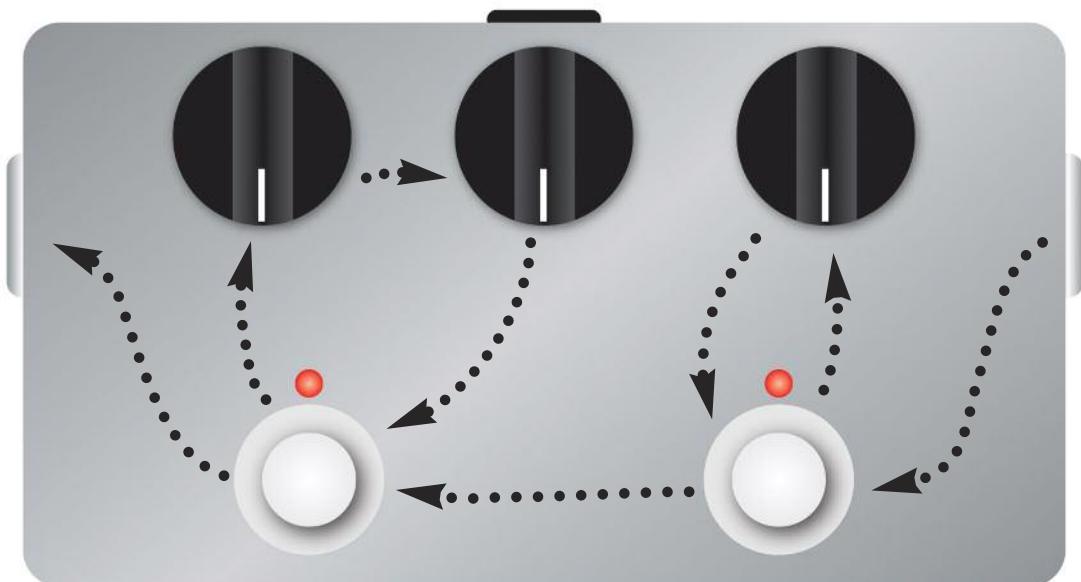
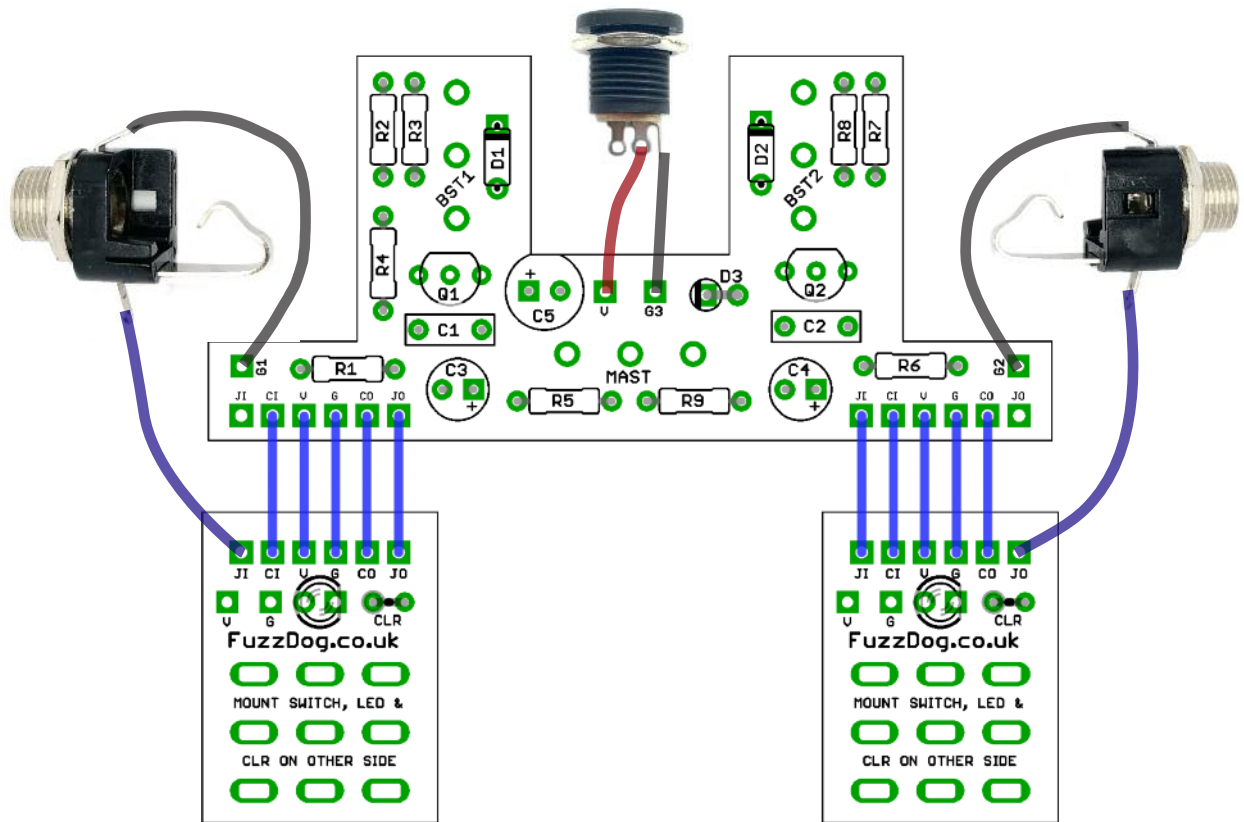
Now drop the daughterboards onto the header pins and footswitches. If drilled precisely as per the template everything should line up nicely. Now tighten the footswitches more securely and solder the headers and switch tags.

Once all is secure, push the LEDs into place in the enclosure and solder.

Final wiring

Your jack grounds can be connected with the G1 and G2 pads on the main PCB.

The jack signal lugs connect to the empty J1 and J0 pads on the daughterboards.



Drilling template

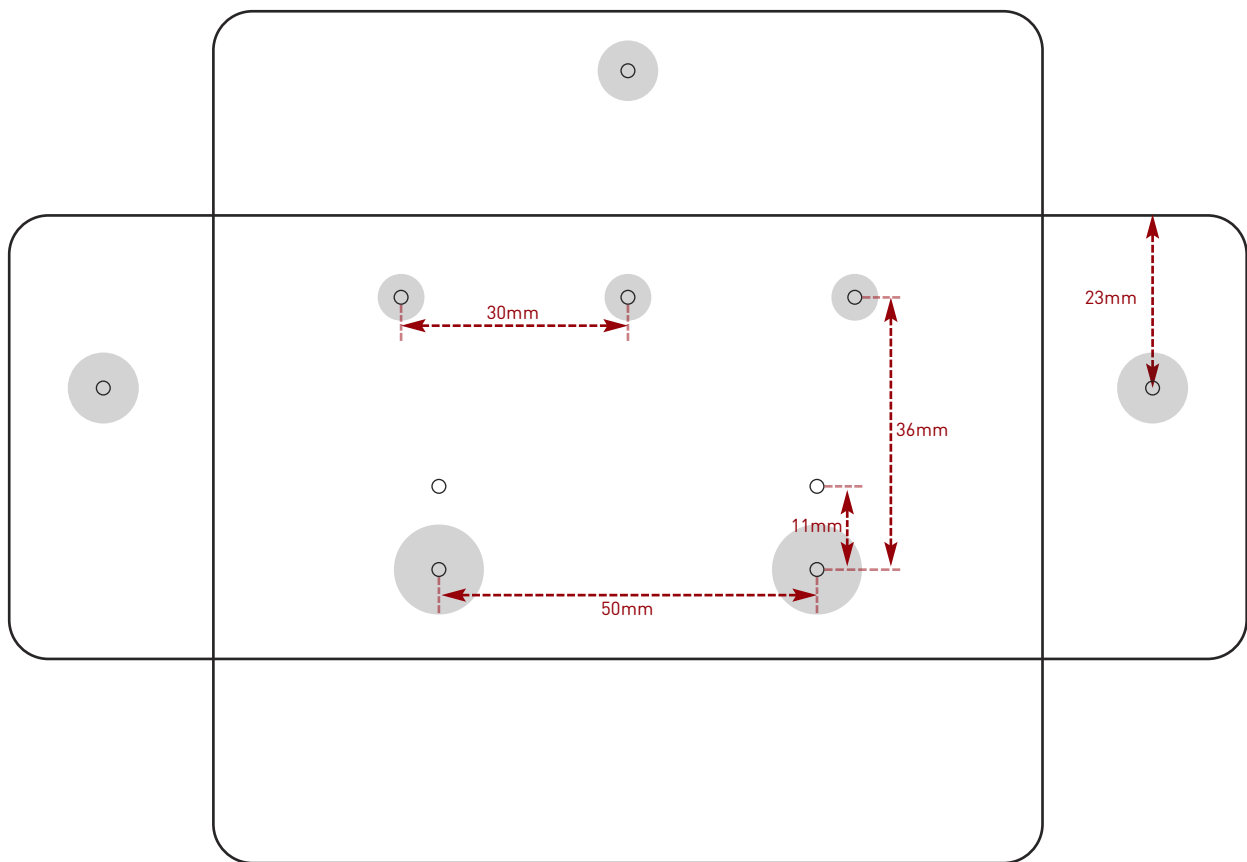
Hammond 1590BB

It's a good idea to drill the pot and toggle switch holes 1mm bigger if you're board-mounting them.

Wiggle room = good!

Recommended drill sizes:

Pots	7mm
Jacks	10mm
Footswitch	12mm
DC Socket	12mm
Toggle switches	6mm



This template is a rough guide only. You should ensure correct marking of your enclosure before drilling. You use this template at your own risk.

Pedal Parts Ltd can accept no responsibility for incorrect drilling of enclosures.

FuzzDog.co.uk

